NUMERICAL PROPULSION SYSTEM SIMULATION

Joseph P. Veres
NASA Lewis Research Center
Cleveland, Ohio

The Numerical Propulsion System Simulation (NPSS) project is managed by the Computing and Interdisciplinary Systems Office at NASA Lewis Research Center. Funding is provided by the High Performance Computing and Communication Program (HPCCP) and by the R & T Base.

The goal of NPSS is to provide a detailed system simulation for use by engine manufacturers to accurately predict performance early during design. Computer simulations are a primary tool used in the design of new turbine engines. A high fidelity system model can quantify the performance of each engine component and its interactions with other components in a system environment. The improved predictive capability introduced into the design process can reduce the number of design and test iterations in an engine development program, and result in lowering the acquisition cost of engines.

Improved performance predictive capability can also benefit the engine system by properly matching components at their peak efficiencies, which in turn can reduce the specific fuel consumption and engine operating costs.

The approach used in the development of NPSS is to combine existing validated computer models for fluid mechanics, heat transfer, combustion, structural mechanics and other disciplines into one large system of codes (Figure CD-96-73883). The NPSS architecture is designed to run the various disciplinary codes in a common simulation environment. The computationally intensive multidisciplinary simulations are to be run on high performance parallel computing platforms to enable rapid affordable computations of engine aerodynamic performance and operability. NPSS is also sometimes referred to as a "Numerical Test Cell for Aerospace Propulsion Systems"

The method used to develop NPSS is a building block approach which three main elements; the Simulation Environment, Engineering Applications and High Performance Affordable Computing (Figure CD-96-73749).

Activities under the Simulation Environment element are focused on the development of a modular architecture for the various disciplinary codes. The National Cycle Program (NCP) is aimed at developing the initial framework for NPSS and is currently focused on the aerothermodynamic cycle simulation process. Toolkits for Configuration Management and the Steady State Cycle Deck Launcher are also being developed under the Simulation environment element. Standards for data exchange are also being established, as well as libraries and utilities.

The High Performance Affordable Computing element of NPSS focuses on computing testbeds that consist of workstation clusters, and efficient parallel communication software. The performance of distributed network of workstations can be at levels that are many times more powerful than a Cray C90. Another activity under this element is the parallelization of engine codes in order to run efficiently on the networked workstations.

Under the Engineering Applications element of the NPSS project, high fidelity gas turbine engine simulations are created that push the envelope of current design practice and computing capabilities. The purpose of the high fidelity simulations is to anticipate the types of computing that will be done in industry routinely during early phases of the engine design process, as computing capabilities become more affordable.

The Engineering Simulations are summarized in the Roadmap for NPSS Simulations shown in Figure CD-96-73364.

As mentioned earlier, the National Cycle Program (NCP) is an important element of NPSS, since it will establish many of the standards to enable communicating (zooming) with the higher fidelity simulations. The thermodynamic cycle model of the engine system is typically used throughout the development of engines, from conceptual design to certification testing. In the NCP system model, each component is represented by its individual overall performance characteristic map. In the case of a new component with no prior test history, the component performance is derived from a database and is a first approximations that ultimately requires validation with a component test rig, or a higher fidelity simulation that does not rely heavily on empiricism. The first version of NCP has been delivered to industry in FY97.

One of the activities illustrated on the NPSS Roadmap is the 3D flow simulation of a complete Low Pressure Subsystem Simulation, coupled to a thermodynamic cycle simulation of the core engine (Figure CD-96-73744). The goal of this project is to numerically simulate the aerodynamic flow in the complete low pressure subsystem (LPS) of a gas turbine engine overnight on a parallel computing testbed. The model solves the three-dimensional Navier-Stokes (N-S) flow equations through all of the components within the low pressure subsystem as well as the external flow around the engine nacelle.

The three-dimensional (3D) N-S flow code for the LPS simulation is the Advanced Ducted Propeller Analysis Code (ADPAC). The LPS modeling project is being done under contract with Allison Engine Company. The engine geometry is the Energy Efficient Engine (EEE), designed by General Electric Aircraft Engines. The 3D flow model of the LPS provides a "tightly coupled" aerodynamic analysis that captures the quasi-steady interaction effects between the components of the LPS. The high pressure core engine is simulated with a thermodynamic cycle code that has been validated with test data. The LPS flow model is linked to the core engine model by sharing common boundary conditions. The 3D LPS model and the thermodynamic model of the core engine together form a "hybrid" flow model of the complete gas turbine engine. Quasi-steady-state operating conditions are simulated iteratively by adjusting the rotative speed of the shaft, until the torque required to drive the fan equals the torque produced by the turbine. The shaft is in equilibrium when the torques are balanced at a quasi-steady-state operating condition. The shrouded rotor labyrinth seals and cavity of the low pressure turbine has been modeled in a tightly coupled simulation with the primary flow. The result of the high fidelity turbine and coupled cavity simulation is an improved match with the data near the shroud endwall region.

The interactions between components within the LPS, as well as the interaction between the LPS and the core engine have been evaluated at the design point condition (takeoff), and at the altitude cruise condition. The large scale (74 block / 6.7M grid point) computer simulation of the LPS has been run in parallel on networked workstation clusters at NASA Ames and on LACE (Lewis Advanced Cluster Environment) at Lewis Research Center.

The N-S modeling of the large low pressure subsystem can provide detailed knowlege of the interaction effects between engine components. The hybrid engine model can be used to evaluate the interaction between the LPS components, while considering the lumped-parameter performance of the core engine. Critical engine component and subsystem interactions can be quantified early in an engine design phase, reducing the quantity and costs associated with hardware testing during an engine design program. The LPS model may be coupled to an external aerodynamic simulation of the airframe to capture detailed propulsion / airframe integration effects on vehicle performance.

The next major element on the Roadmap for NPSS Simulations of Figure CD-96-73364 is the 3D Flow Simulation of the Full GE90 Turbofan Engine detailed in Figure CD-96-73759. This project leverages from ongoing efforts between General Electric and NASA in developing NASA's average passage flow code (APNASA) and workstation clustering technology. The full engine simulation task is done in cooperation with General Electric Aircraft Engines under the Large Engine Technology contract and at NASA Lewis Research Center. The APNASA model of the GE90 engine utilizes a simplified 2D model of the combustor, that obtains source terms from a combustor simulation. The GE90 turbofan simulation models the primary flowpath of the two spool engine containing 50 blade rows. Inflow and outflow bleeds are modeled as source terms to the APNASA primary flowpath flow model. A valuable part of the project is the establishment of a common geometry model of the GE90, and the flow of data between the numerous groups involved in its design. The project is done in phases, with the initial phases focusing on engine component simulations and validation with test rig data. A multi block version of the APNASA code is also being developed by GE as part of the GE90 modeling project. The cooled high pressure turbine and low pressure turbine simulations have been initiated, with further validation runs to be done in the next fiscal year. The high pressure compressor simulation is also proceeding well and will continue through the fiscal year.

The large scale simulation of the GE90 engine, together wit the National Cycle Program help to define the requirements for the computer architecture and simulation environment for the NPSS project.

The next element illustrated on the Roadmap for NPSS Simulations of Figure CD-96-73364 is the Axisymmetric Engine Simulation project, summarized in Figure CD-94-70625. This task uses an axisymmetric Euler flow model to simulate the complete Energy Efficient Engine. Other engines that have been simulated by the axisymmetric model include a conceptual version of the High Speed Civil Transport engine. The performance of the engine is simulated by running external component design codes and using their output as source terms in the axisymmetric engine system model of the complete engine at the design point. An axisymmetric engine model can be utilized during the design process to size the flowpath through the turbomachinery, as well as the flowpath through the bypass and bifurcated ducts, and the jet nozzle. This model can predict the performance characteristics of each component in the engine with improved accuracy over the thermodynamic cycle estimates. The 2-D model can result in improved estimates of the flow conditions near the endwalls of the flowpath, throughout the turbomachinery in the engine system. The 2-D component codes estimate the performance based on an empirical database of losses and flow deviations together with the Euler equation. The source terms from the 2-D design point turbomachinery component codes are transfered as source terms to the 2-D engine system model to obtain an axisymmetric Euler flow simulation of the complete engine. The component codes are empirical based 2-D, or quasi-3-D flow codes with loss models that are validated for off-design values of incidence and loadings. Combustor performance is estimated from a simplified combustor model and specified by source terms into the 2-D engine system model. Inflow and outflow bleeds from the flowpath are also specified by source terms in the component codes and the 2-D engine system model. The effects of disk pumping and other parasytic losses outside the primary flowpath are not modeled at the axisymmetric level.

The next major activity illustrated on the Roadmap for NPSS Simulations of Figure CD-96-73364 is the 3D Reacting Flow Solver for the National Combustion Code (NCC), named CORSAIR_CCD. The goals of the combustion modeling flow solver project are to develop an integrated system of codes that will enable the multidisciplinary analysis of the full combustor from compressor exit to turbine inlet, in an overnight turnaround timeframe. An important element of this project is to significantly reduce the computer turnaround time of a full combustor analysis from 28 days to 1 day, in a phased approach. The combustor modeling project is summarized in Figure CD-96-73761. There is an ongoing effort to improve the parallel performance of the flow solver, and improve the fidelity of the combustion model. Improved message passing techniques within the solver have significantly reduced the communication time between the multiple processors, and increased the scaleability efficiency to better than 96% with 24 processors. Other models that have been added to improve the fidelity of the CORSAIR_CCD solver are accelerated kinetics, fuel spray model, turbulent combustion module, second-order turbulence model and a radiation model. Besides the CORSAIR_CCD flow solver, the complete NCC has other main elements, that together form a complete design system for combustors (Figure CD-96-73691). The pre-processor for the combustor code can accept geometry from a third-party CAD package, and generate an unstructured mesh using PATRAN, or CFDGEOM. The latter has been developed by CFD Research under an SBIR from Lewis Research Center. The NCC package also includes a post-processor package that enables the designer to evaluate key parameters such as pattern factor, temperature and pressure profiles, losses, and NOX emmissions.

Another key element of NPSS is the development of a methodology to enable multidisciplinary analysis of a high pressure ratio compressor. The task outlined in Figure CD-96-72720 summarizes the Coupled-Aerodynamic-Thermal-Structural (CATS) analysis of a multi stage aircraft engine compressor. The effects of geometry variations such as blade deflections and changes in clearances due to thermal and structural effects are modeled iteratively. Variations in blade shapes from the design-point hot running geometry are estimated by iterating between the aero, thermal and structural analysis tools. The MD tools model the interactions between aerodynamic-thermal-structures for improved simulation of the compressor performance. Multidisciplinary compressor modeling results in improved predictions of blade geometry deformations and clearances at "hot running" engine operating conditions. The subsequent aerodynamic analysis of the hot running geometry results in an improved prediction capability of compressor performance and stall margin.

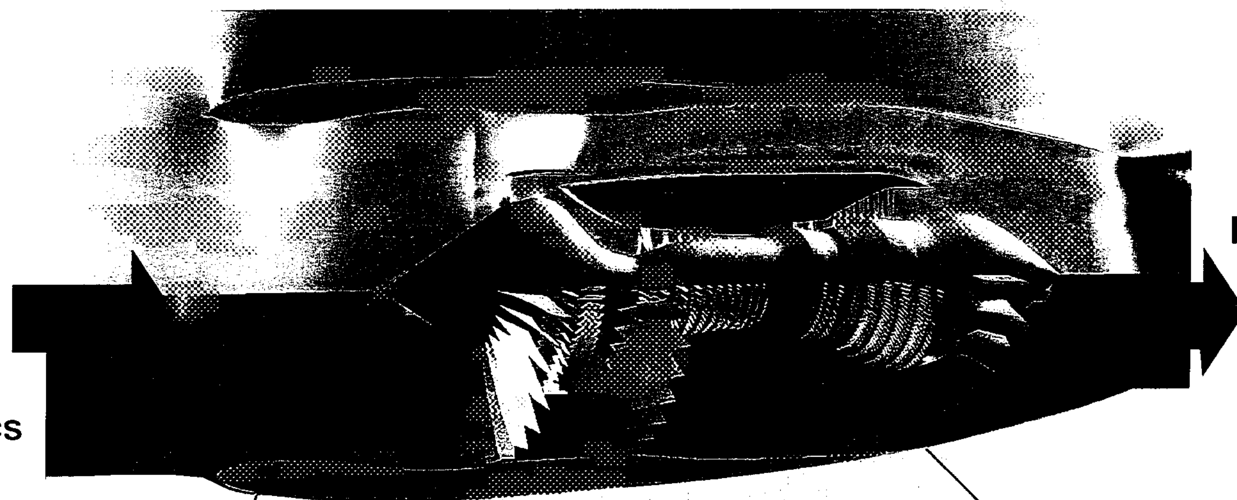
A task to improve the fidelity of simulations near the endwall region is being done under contract with the Allison Engine Company (Figure CD-96-73713). The task numerically investigates the flow details within a compressor stator seal cavity that is fully coupled to the detailed flow model through the stator. The flow code that is used in this task is ADPAC, a 3D Navier-Stokes flow code. Utilizing the ADPAC flow code, the seal cavity flow and the stator flow fields are solved simultaneously by a variety of methods. The first method was a full 3D aerodynamic model of the coupled cavity and stator using time accurate rotor exit conditions upstream of the stator. That simulation was computationally intensive, and simpler, faster methods were investigated that would still maintain modeling accuracy to match the test data. It was found that a 2D stator seal cavity simulation coupled to the 3D steady state model of the stator flow field adequately matched the results obtained by the test data Figure CD-96-73790). This simulation provides improved fidelity in the stator endwall and cavity regions, and is a significantly faster computation that can be used during stator and seal design.

All of the tasks that are detailed above will be integrated in phases into a steady-state 3D aerothermodynamic simulation of a full engine primary flowpath in the 2001 timeframe.

Numerical Propulsion System Simulation (NPSS)

Validated Models

- Fluid Mechanics
- Heat Transfer
- Combustion
- Structural Mechanics
- Materials
- Controls
- Manufacturing
- Economics



Rapid Affordable Computation of:

- Performance
- Stability
- Cost
- Life

NPSS

Integrated Interdisciplinary Analysis and Design of Propulsion Systems

High Performance Computing

- Parallel Processing
- Expert Systems
- Interactive 3-D Graphics
- Networks
- Database Management Systems
- Automated Video Displays

A Numerical Test Cell for Aerospace Propulsion Systems

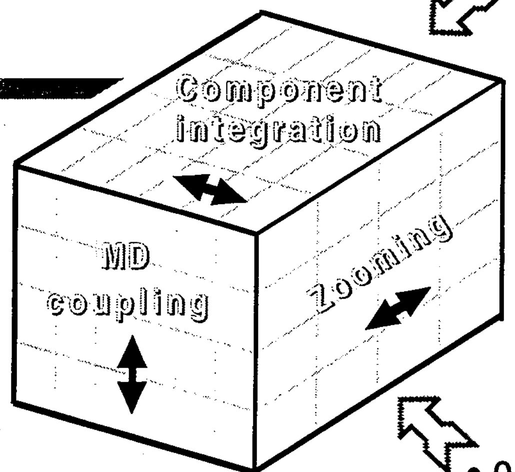
CD-96-73883

NPSS
Work Breakdown Structure

Simulation Environment

- Modular Architecture
  - — National Cycle Program
- Toolkits
  - — SSCD
  - — CM Manager
- Library/Utilities
  - — NURBS
  - — PEV
  - — Data Standards

Engineering Applications

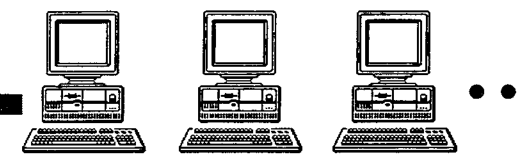


- National Cycle Program
- Axisymmetric Engine
- 3-D Subsystems/System

- Coupled Aero-Thermal-Structural (CATS)
- CFD/Controls
- Spectrum
- VCE
- MSAT

- 0-D Engine/ 1-D Inlet
- 0-D Core/3-D LP Subsystem
- 2-D Combustor/ 3-D Engine

High-performance, Affordable Computing

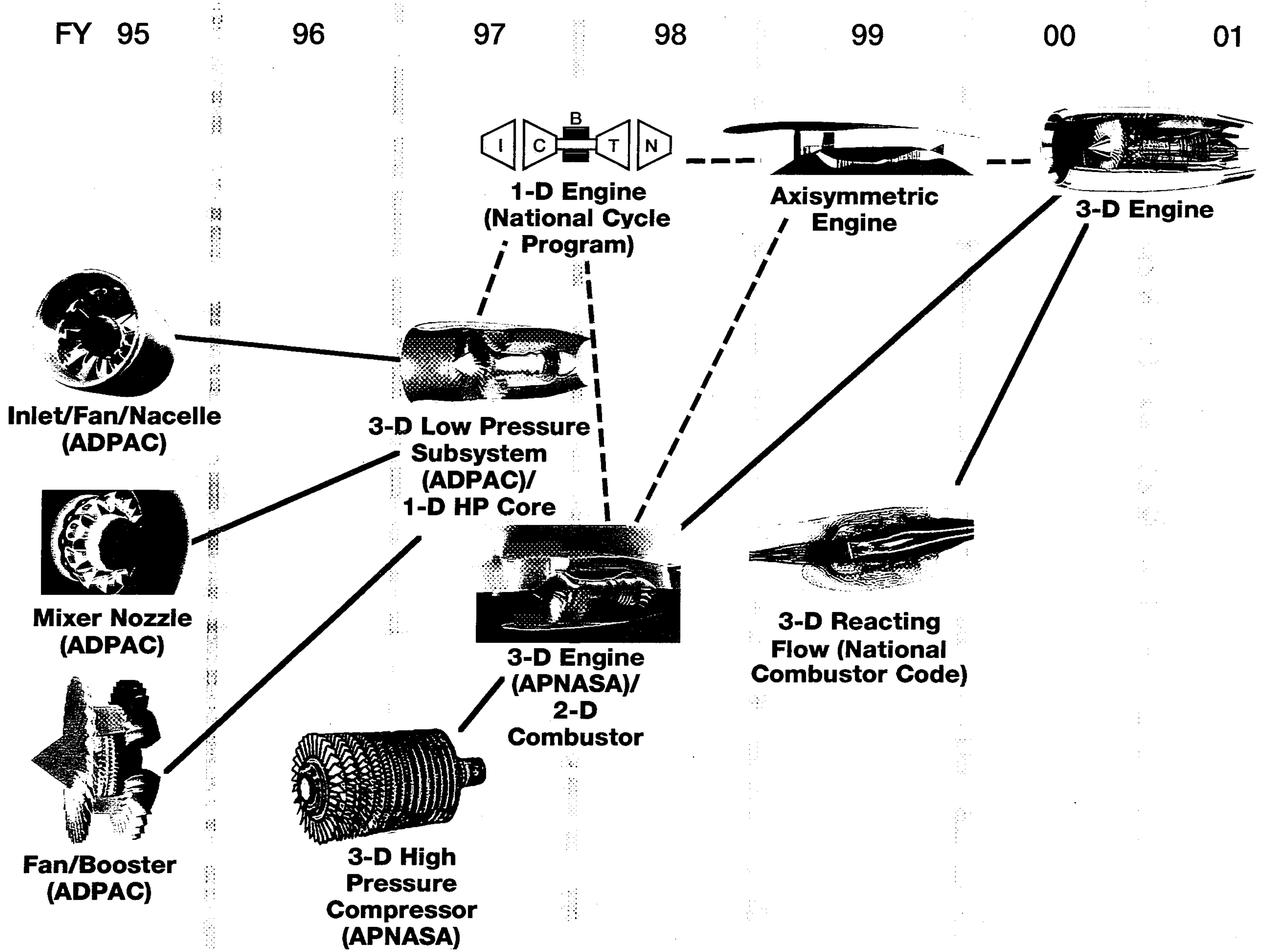


- Clustered Workstations
- High-Speed Networks
- Parallel System Software
- Code Parallelization

Computing and Interdisciplinary Systems Office
Lewis Research Center

CD-96-73749

Roadmap for NPSS Overnight Simulations
(Aerothermodynamic/Primary Flowpath/Steady-State)

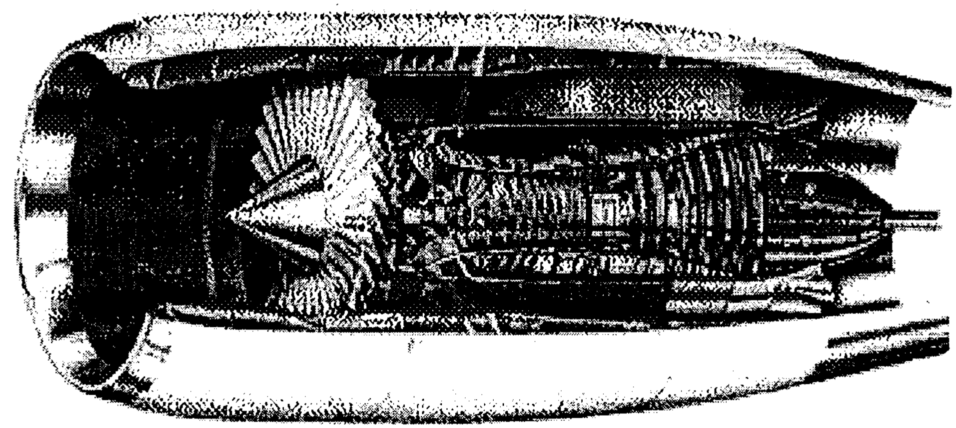


CD-96-73364

Low Pressure Subsystem 3-D Model

Objective

Develop a detailed flow simulation of the low pressure subsystem within a gas turbine engine using a simplified core engine model.



Approach

- Apply the ADPAC 3-D Navier-Stokes flow code to the Energy Efficient Engine LP Subsystem consisting of: nacelle, inlet, fan, bypass duct, mixer, LP turbine and nozzle creating a 3-D flow model of LP Subsystem
- BC's at the core engine inlet and exit will be specified from thermodynamic cycle and data
- The simulation will run in parallel on distributed workstations

Impact/Metrics

- Evaluate the interaction effects between the LP subsystem components while considering the boundary conditions at the core engine
- The LPS model will reduce design/development time by enabling the designer to numerically investigate engine operability

Applications

- HSR High Speed Civil Transport engine
- AST engines to assist in the developmer and certification of growth versions of existing engines

Point of Contact

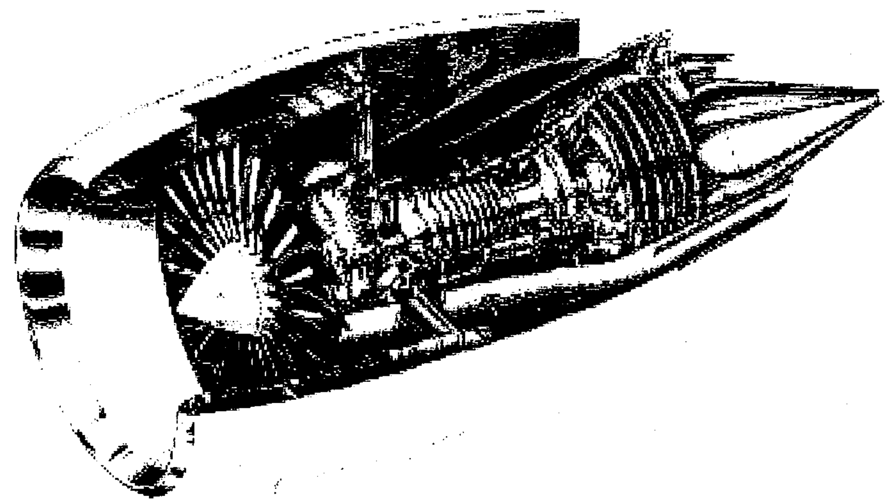
Joseph Veres
tel.: (216) 433-2436 fax: (216) 433-518
e-mail: jveres@lerc.nasa.gov

CD-96-73744

HPCCP/NPSS Plan Detailed Flow Simulation of Modern Turbofan Engine

Objective

Develop a detailed flow model of a full turbofan engine that runs on parallel workstation clusters overnight. The model will simulate the flow in the primary flowpath andwill have a simplified combustor model.



Approach

- ✦ The 3-D flow analysis will model the GE90 turbofan engine using APNASA (NASA's average passage flow code)
- ✦ The project will leverage from current efforts between NASA and G.E. in developing the APNASA flow code and workstation clustering technology

Significance/Metrics

The overnight 3-D flow simulation capability of the primary flowpath in a complete engine will enable significant reduction in the design and development time of gas turbine engines.

Point of Contact

Joseph Veres
tel.: (216) 433-2436
fax: (216) 433-5188
e-mail: jveres@lerc.nasa.gov

CD-96-73759

NPSS Demonstrates Axisymmetric Engine Simulation Coupled to Compressor Design Code

Objective:

Develop the capability to computationally evaluate the interactions between components and full engine.

Approach:

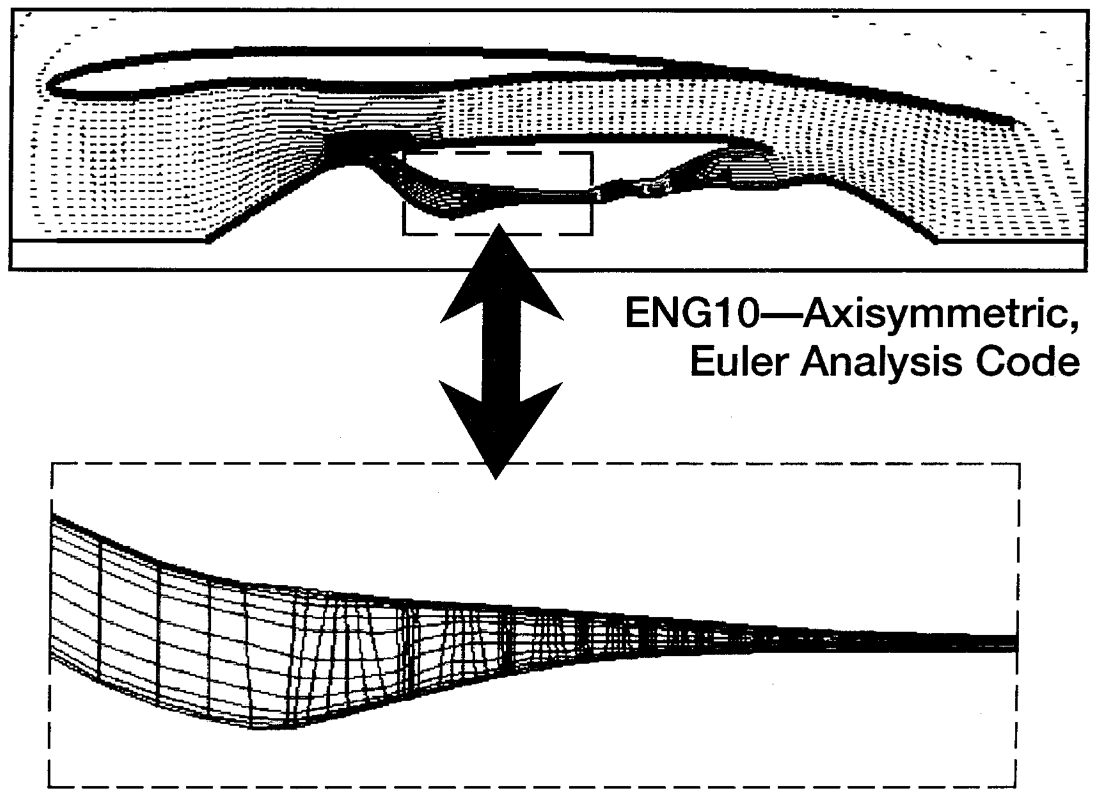
Components are modeled in the engine simulation as source terms computed by the design code. Engine/component interactions take place through boundary conditions.

Impact:

Reduce engine development time and cost by enabling evaluation of component designs including the full engine interactions early in the design cycle.

Point of Contact:

Isaac Lopez — NASA LeRC



ENG10—Axisymmetric, Euler Analysis Code

PERCH—GEAE Streamline Curvature Compressor Design Code

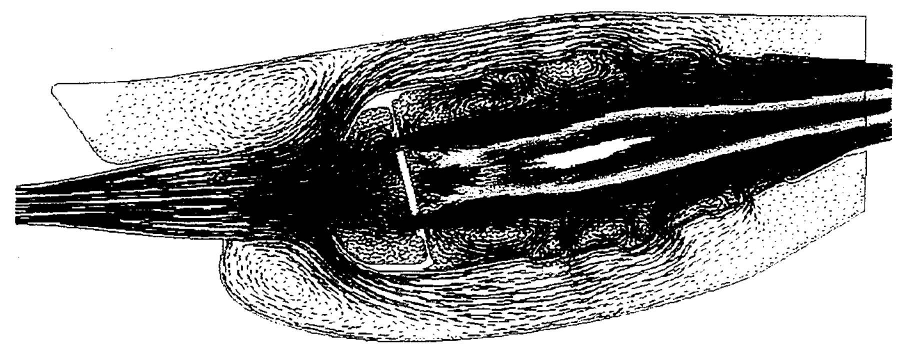
CD-94-70625

Flow Solver for National Combustion Code

Multidisciplinary Combustor Design and Analysis System with Emissions Modeling

Objective

Develop an integrated system of codes for combustor design and analysis to enable a factor of 5 reduction in cost and analysis time



Approach

- ✦ Develop a computational combustion dynamics capability (CCD)
- ✦ CORSAIR-CCD is a Navier-Stokes flow solver based on an explicit four-stage Runge-Kutta scheme
- ✦ Unstructured meshes
- ✦ Run on networked workstation clusters
- ✦ The solver can be linked to any CAD system via the Patran file system

Significance/Metrics

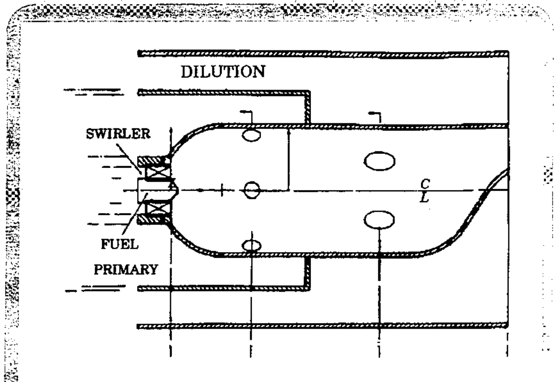
- ✦ The National Combustor Code is a system of codes that will enable the multidisciplinary analysis of the full combustor from compressor exit to turbine inlet
- ✦ The CORSAIR-CCD code is the baseline flow solver module

Point of Contact

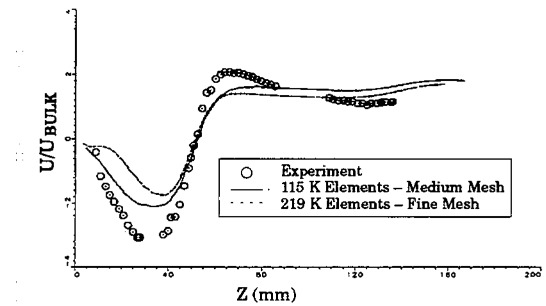
Dr. Nan-Suey Liu
tel.: (216) 433-8722 fax: (216) 433-5802
e-mail: fsliu@lerc.nasa.gov

CD-96-73761

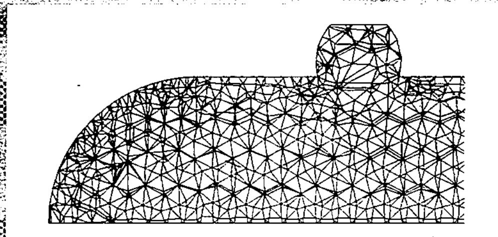
National Combustion Code



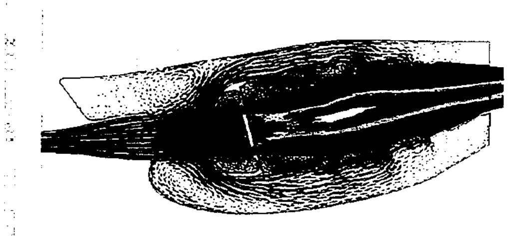
HPCCP Demonstrates 5X Speedup of Combustor Analysis Time on HSCT Design Concept: LDI/MVS



Design
(CAD)
- Geometry Modeling
- Commercial CAD Software



Pre-Processing
(PATRAN/CFDGEOM)
- CAD File Cleanup
- Unstructured Mesh Generation
- Full Combustion System



Analysis
(CORSAIR)
- Improved Parallel Performance: 96% Efficiency/24 Processors
- Better Communication
- Accelerated Kinetics
- Spray Model
- Turbulent Combustion Module

Post-Processing
(CORPERF/PV3)
- Design Parameters
- Pattern Factor
- Radial Temperat. Profile
- Temperature Rise
- Pressure Loss
- NOX Post-Processor
- PV3 Visualization

2X Speedup

2.5X Speedup

Overall Speedup of 5X

CD-96-73691

HPCC/NPSS Develop Interdisciplinary Analysis of Aircraft Engine Compression System

CATS—Coupled Aerodynamic-Thermal-Structural Analysis

Objective

Streamline multidisciplinary simulation process for aircraft engine compression systems

Approach

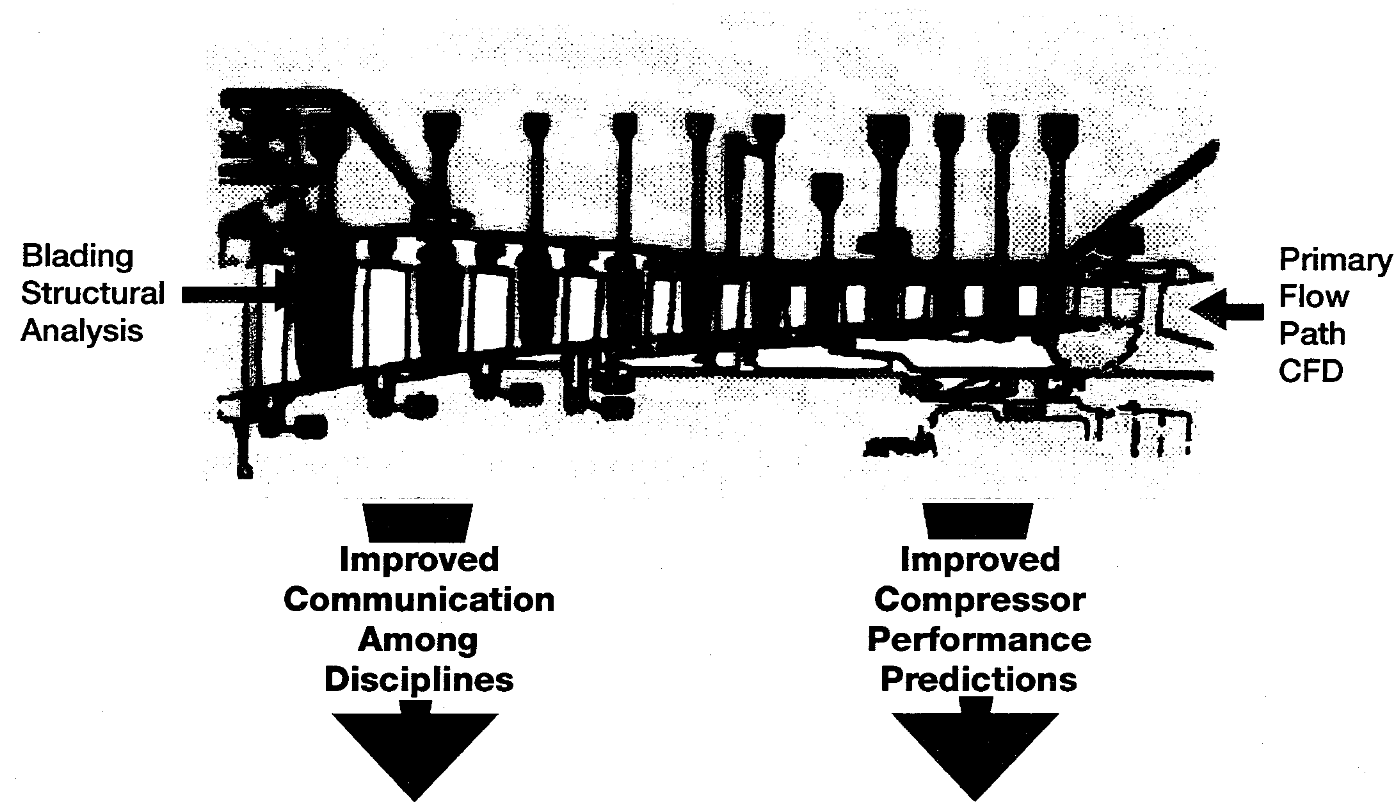
Integrate the aerodynamic, thermal and structural computational analyses using data management and NURBS based data mapping

Impact

Reduce compressor design time/cost and improve performance

Point of Contact

Chuck Lawrence, NASA LeRC



Stator Seal Cavity Investigation

Objective

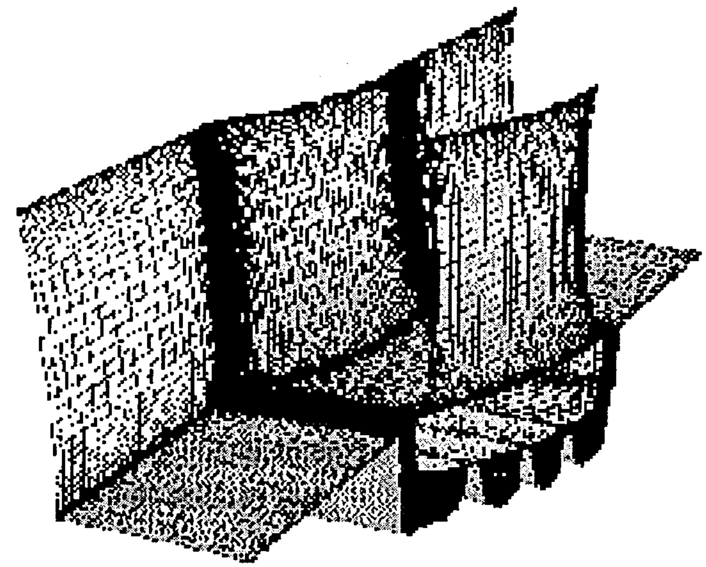
Numerically investigate the flow details within a compressor stator seal cavity that is fully coupled to the detailed flow through the stator.

Approach

- ✦ Use the ADPAC code to solve the seal cavity flow and the stator flow fields simultaneously
- ✦ Perform a parameterized study of seal cavity geometry on stator performance
- ✦ The geometry is an inner banded stator seal cavity from an advanced multistage axial compressor (AST candidate). Geometric variations include seal tooth gap, cavity gap, seal cavity volume, stator land radial mismatch and wheel speed

Point of Contact

Dr. Michael Hathaway
tel.: (216) 433-6250
e-mail: mhathaway@lerc.nasa.gov



Significance/Metrics

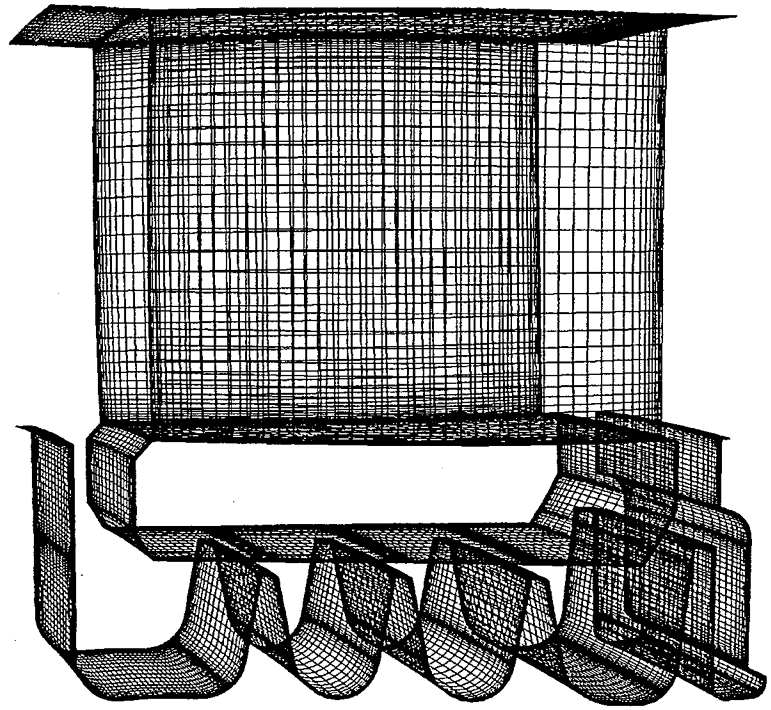
- ✦ First reported analysis of coupled cavity/stator flow field
- ✦ Seal exit tangential momentum plays a critical role in rotor performance, stator incidence, and the suction side separation near the hub
- ✦ Grid resolution studies show that 500K mesh points are required to resolve the tightly coupled stator/seal cavity 3-D flow fields

CD-96-73713

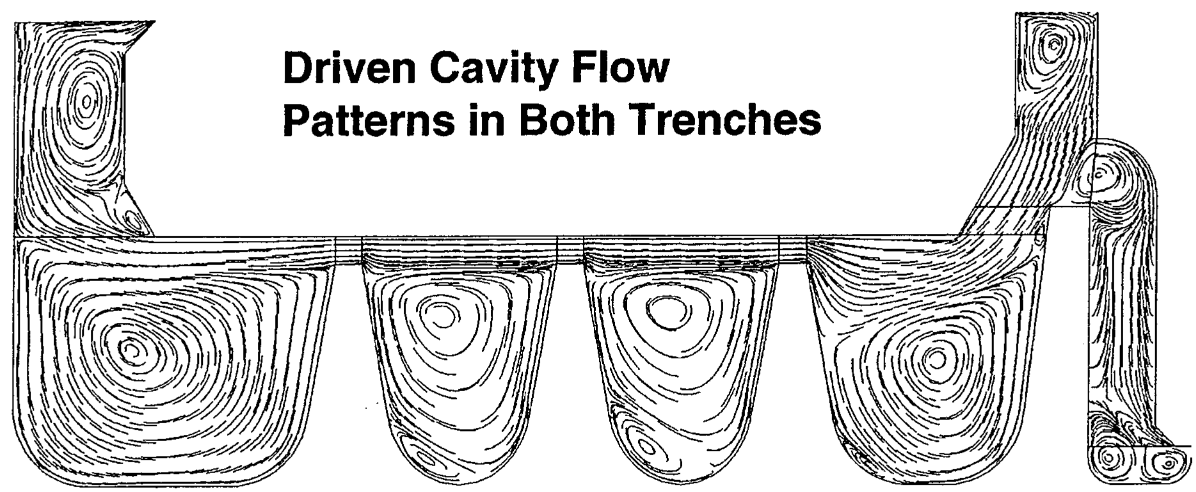
Stator Seal Cavity Flow Investigation

Tools Used:

TIGG3D / GRIDGEN



Triple–Knife Seal Cavity



CD-96-73790